Numerical Prediction of an Axial Turbine Performance Used for Automotive Engines Turbocharger

Mohsin Obaid Muhi

Department of Mechanical Techniques ,Technical Institute of Karbala

Al-Furat Al-Awsat Technical University, Iraq

mohsinaltaai@yahoo.com

Abstract

This paper investigate numerically the flow inside an axial turbine of turbocharger in the three dimensions. The performance of an axial turbine for the turbocharger is heavily affected by blade design due to the flow dynamics in the blade passage. Furthermore, modification and improvement of a turbine blade is a challenging task for turbomachinery engineers. Hence, this study is aimed to predict the performance of an axial turbine of automotive engine turbocharger. A cascade, having nozzle stator, was used for the performance evaluation. A commercial CFD code ANSYS CFX, was used to simulate the flow and the primary grid generation. The influence of blade numbers of rotor and stator has been studied by comparing the turbine stage performance. The results showed that the fluid dynamics within an axial turbine has indicated that the pressure ratio and volume flow rate were predicted. Obviously, it was observed that the pressure ratio, volume flow rate and efficiency were predicted numerically. The comparative study of these results indicated noticeable a turbine stage performance. Overall, numerical results obtained from computational fluid dynamic simulations could produce a much deeper understanding of flow and a highly reliable for estimation on the performance of an axial turbine of turbocharger.

Keywords: turbocharger; axial turbine; stator nozzle; CFD.

الخلاصة

في هذا البحث تم اجراء تحقيق عددي في تدفق ثلاثي الأبعاد داخل توربين محوري لشاحن توربيني. اداء التوربين المحوري للشاحن التوربيني يتأثر بشكل كبير بتصميم ريش التوربين بسبب التدفق الديناميكي في ممر الريشة. وعلاوة على ذلك، تعديل وتحسين ريشة التوربين هو مهمة صعبة لمهندسين المحركات التوربينية. وبالتالي،هذه الدراسة تهدف إلى تنبؤ أداء التوربين المحوري لمحركات السيارات ذات الشاحن التوربيني. سلسلة الريش تحتوي على فوهة ثابتة استخدمت لتقييم الأداء. تم استخدام كود تجاري من CFD انسز CFX لمحاكاة تدفق الجريان وانشاء الشبكة الاولية . وقد تمت دراسة تأثير عدد ريش الجزء الدوار والجزء الثابت من خلال مقارنة أداء التوربين. وأظهرت النتائج أن ديناميكية السوائل داخل التوربين المحوري بينت ان نسبة الضغط و معدل حجم التدفق والكفاءة تم توقعها. لوحظ أن نسبة الضغط، معدل حجم التدفق والكفاءة تم توقعها عدديا. أشارت الدراسة المقارنة لهذه النتائج بشكل ملحوظ لأداء التوربين. وعموما، أن النتائج العددية التي تم الحصول عليها من المحاكاة الحسابية لديناميكية السوائل يمكن ان تقدم فهم اكثر للجريان وموثوقية عالية لحساب أداء التوربين المحوري للشاحن التوربيني.

الكلمات المفتاحية:– شاحن توربيني ، توربين محوري ، شفرات ثابتة موجهة ، الحسابات الديناميكية للسوائل .

Nomenclature

CAD	Computer Aid Design
IGES	Initial Graphics Exchange Specification
TE	Trailing Edge
LE	Leading Edge
Pt	Total Pressure
Ps	Static Pressure
Tt	Total Temperature
Ts	Static Temperature
Mabs	Absolute Mach Number

Mrel	Relative Mach Number
Cm	Meridional Velocity
R	Blade Radius Location
Z	Blade Axial Location
k-ω-SST	K-Omega Turbulence Model
k-ε	k-Epsilon Turbulence Model
3D	Three Dimensions
2D	Two Dimensions
NACA	National Advisory Committee for Aeronautics

1. Introduction

Nowadays, turbocharging is playing a fundamental role not only to improve automotive engine performance, but also to reduce fuel consumption and exhaust emissions for both spark ignition and diesel engines**.** Small turbocharger turbine designs are usually based on radial or mixed-flow type when they are connected to small engines, such as car or truck engines. In large marine diesel engines, axial turbines are mostly used.The interest has more increasingly been devoted to the development of turbocharger because of their compact size, large capacity, high performance, and ability to improve volumetric efficiency. The improvement of turbocharger of axial turbine performance and extension of the stable operating ranges are becoming critical for the viable future of low emission engines. It is widely recognized that turbocharging is a very useful technique commonly applied in both spark-ignition and compression ignition engines to improve engine efficiency.

Thus, the effects of changing the flow conditions at the inlet to a mixed flow turbine have been studied and the results have showed that the inlet flow angle significantly influenced the degree of reaction across the rotor and the turbine efficiency (Morrison *et.al.,* 2016).Characteristics of a two-stage turbine on a turbo-compound engine have been investigated numerically under pulsating flow conditions so as the behaviors of turbine stages were studied under low, mid and high load conditions. Results indicate that the Low pressure turbine is more sensitive to the pulsating flow, especially at low load conditions, compared with high pressure turbine (Zhao *et.al.,* 2016).A method was presented to evaluate the transient performance of a turbocharger which contains the speed fluctuations of the turbocharger affected by the exhaust pulsations in addition to the change in mean speed during the load step**.** Results observed that the time scale of the exhaust pulsations is smaller than the change in mean speed of the turbine (Roclawski1 *et.al.,* 2016). Nozzle vanes were added as an integral part of the stator design to reach better flow guidance into the turbine blades.The comparison were studied to detect changes in the circumferential flow angle distribution between both volute arrangements under steady and pulsating flow operating conditions. The results indicated that there are significant differences in the flow angle behavior for both volutes regardless of the flow conditions(steady or unsteady) (Padzillah *et.al.,* 2014).To realize high compression ratio, engine equipped with two-stage turbocharging system is used as an effective way which increase engine power density and efficiency of exhaust gas energy. The static pressure field at inlet to low pressure turbine increases back pressure of high pressure turbine , however efficiency of high pressure turbine changes little which appeared at the calculation results (Liu *et.al.*, 2016).

However, to measure the widest turbine performance map an experimental study on a turbocharger turbine with different techniques was used at different turbine inlet temperatures which widened the measuring area. Another technique was also used to introduce air at high pressure at the compressor inlet increasing thereby its power consumption and expanding the measuring area. The turbine efficiency including the mechanical losses and the turbine total-to-static isentropic efficiency were calculated for the turbocharger (Salameh *et.al.*, 2016). A small size turbocharger typically working in automotive economized engine applications is carried out experimentally using an innovative hot gas generator system. By comparison with the experimental results obtained to a previously published work (Bontempo,2014**),** several enhancements in the rig actual configuration layout and in the measuring chain are existing of the performance of a turbocharger (Bontempo *et.al.*, 2015). Due to the difficulty of a correct evaluation of the turbine outlet temperature, the direct evaluation of isentropic efficiency at the inlet and outlet sections gives a significant errors so that a primary experimental analysis was done. In actual, a specific “hand-made” three holes probe was adopted to measure thermodynamic quantities downstream the turbine and flow field distribution (Marelli *et.al.*, 2016). Different temperature and pressure of exhaust gas were studied in experimental set up to investigate the performance of turbocharger. Workbench platform of Ansys CFX is used to study the effect of exit angle of turbine blade .The results showed that the speed of turbine shaft increases at certain level and after that it will remain constant as the engine load increases (Patel and SUBHEDAR ,2016).

Therefore, a number of mixed flow turbine designs were studied by using the CFD model. These involved studies changing the cone angle and the associated inlet blade angle. The results of this study offer insight into the performance of a mixed flow turbine with respect to cone and inlet blade angle (Leonard *et.al.*, 2013).Opening and closing of the exhaust valves, further deteriorates the flow angle distribution around its circumference so that to offer improved on the flow angle behavior, flow angle swings over a wide range (300% more than the steady state condition) in pulsating flow field . It was also found that the flow angle fluctuations during pressure drop period is significantly lower as compared to that during pressure increment period by (19) (Padzillaha *et.al.*, 2015). Generation of strong secondary vortices in an axial turbine stage caused by the flow through a radial gap above and below ends of prismatic blades which significantly affect an efficiency of the stage (Straka, 2016). Four linear cascades of turbine blades were studied both experimentally and numerically. The results delivered insight into the role of the base pressure in the profile losses over the transonic regime. It was concluded that an accurate prediction of the base pressure may serve as a basis for a revised Mach number correction to be applied to the profile loss correlation in the transonic and supersonic flow regimes (Kibsey and Sjolander , 2016).

Moreover, constructing a 3-D model with four discontinuous equally-distributed nozzle blocks. By investigation the influence of admission modes ,two partial admission modes (A&B) were analyzed separately and compered with the full admission situation (C) .The results showed that partial admission could cause extra mixture losses and lead to lower efficiency, full admission (Mode C) performed best in efficiency, and Mode B performed better than Mode A under partial admission conditions (Pan *et.al.*, 2016). A new technique that integrates the modeling of an efficient small scale subsonic axial turbine with ORC cycle analysis at low temperature heat sources using many of organic working fluids. Results showed that, the maximum isentropic efficiency is 82% and power output 5.66 kW leading to cycle efficiency of 9.5% by using working fluid R123 for a turbine with mean diameter of 70mm (Al Jubori *et.al.*, 2016). After a series of design, the final configuration of single stage highly loaded axial turbine is analyzed by using commercially available RANS CFD software and the design is carried out by aiming 88% total-to-total efficiency. Thorough 3D-RANS CFD study of the turbine shows that, the design requirements of turbine are reached with enhanced efficiency of 90%. (Rajeevalochanam *et.al.*, 2016).Turbine blade vibrations were studied in this project and describes practical examples of the application of unsteady computational fluid dynamics (CFD) and finite element methods (FEM) applied to the development of axial turbines. The results show that the use of modern numerical methods reduces cost and required time in the design of axial turbocharger turbines as well as reduce the experimental effort (Filsinger *et.al.*, 2005). The dynamic loading of the turbine blades can be determined in the design process by using this coupled CFD - FEM analysis. The time resolved two-dimensional coupled stator - rotor flow field of the turbine stage was calculated, To predict the excitation forces acting on the rotating blades. (Filsinger *et.al.*, 2001).

In the light of the facts that given above, the work reported in this paper deals with numerical investigations on an axial turbine of turbocharger ,were CFD simulations and flow behaviour of an axial turbine were performed. In order to reduce cost and required time to design and redesign an axial turbine used for automotive engine as well as reduce experimental effort to get rid the aerodynamics problems ,this study is focused to predict the performance of an axial turbine of automotive engine turbocharger. A cascade, having nozzle stator was used for the performance evaluation.

2- Specification & Design

Once the blade to blade design procedure has been completed, a drawing, both in 2D and 3D, is made. The 3-D geometry of the axial turbine figure 2 is created using a dedicated commercial software (ANSYS Blade modeler) .The variation of the beta angle (β) along the rotor is established by a spline curve for every layer (mid, hub and shroud) that can be configured and modified in the program. The blade profile chosen for both, the rotor and stator, is a general NACA profile. This study has an axial turbine with nozzle stage model. The inflow and outflow of the fluid zone are shown in figure 1. The main geometry features and dimensions of an axial turbine are given in table 1.

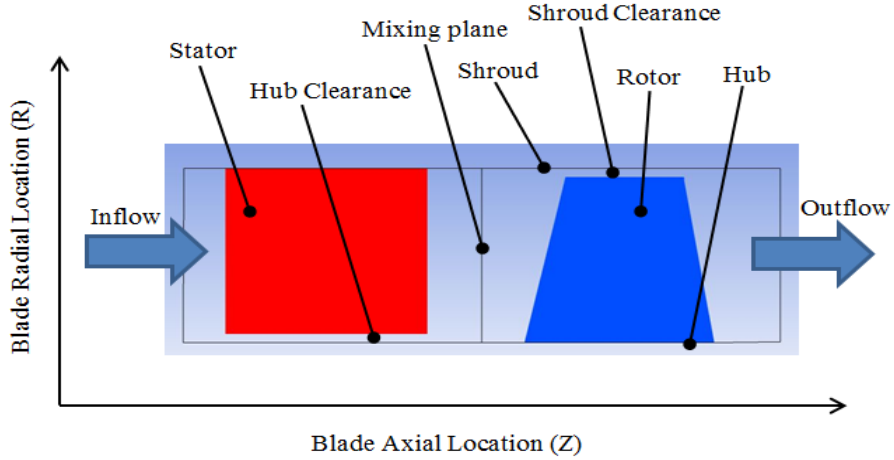


Figure 1: Rotor and stator on the Meridional View(Ansys Blade Modular)

Table 1: Axial Turbine (Rotor and Stator) Features.

Axial Turbine Geometry	Rotor
Blade Axial Chord at the Hub	32 mm
Blade Height	26 mm
Rotating Speed(RPM)	30000
Outer Radius of Impeller	76.37 mm
Blades Number	35
Shroud Tip Clearance (% span)	95%
Nozzle Geometry	Stator
Blade Number	20
Blade Axial Chord at the Hub	32
Hub Tip Clearance(%span)	95%

Figure 2 shows the geometry of an axial turbine wheel stator nozzle stage comprising of 35 main rotor blades and 20 stator blades. The CFD computations for the design are performed on the geometries. All the surface geometry, inlet, exit, and periodic boundaries, were defined via CAD as IGES parts.

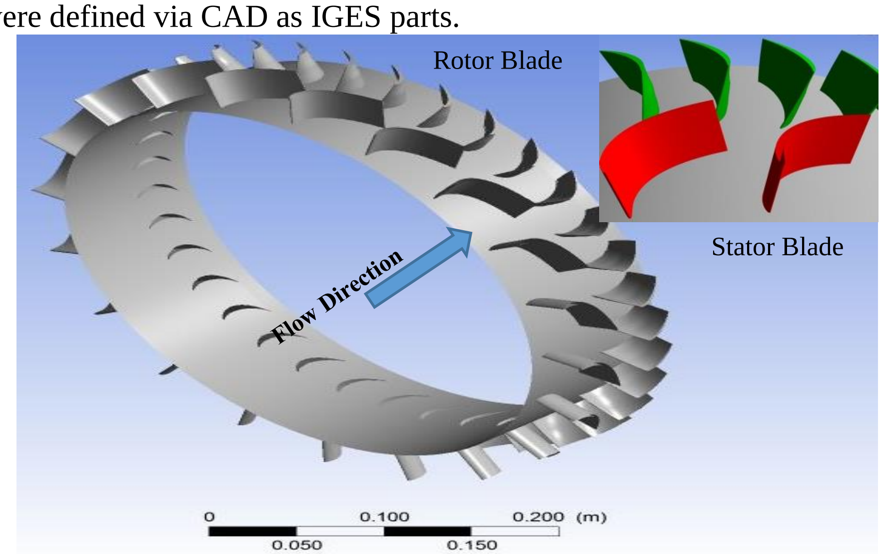


Figure 2: Three Dimensional View of an Axial Turbine stage

3. CFD Methodology
3.1 Grid Generation

A commercial CFD code is used to carry out the numerical study. The fluid domain is extracted for the rotor blade to blade passage with the stator blade itself as the inlet guide domain .The surface mesh is generated by using a hexahedral mesh on the rotor and stator surfaces. The surface repair tools have sufficient control to allow the analysis by choosing among the components to include and exclude in the meshing. This is to control the size of mesh in various parts by using the surface curvature or by defining local refinement zones. Once these surface mesh control settings are defined, the tool retains the association with the imported CAD parts. This makes the parametric modelling of the components very easy. The finite element mesh is generated by using hexahedral, as validated for flow and thermal solutions. The hexahedral element mesh consists of 8 nodes, agglomerated from the underlying automatically generated hexahedral mesh. These meshes offer significant advantages over traditional mesh types (Layth *et.al.,* 2014). The computational grid generated for the rotor blade to blade passage comprising exactly 406179 structured hexahedral elements in multi-block environment, and that for the stator nozzle passage comprising 488074 hexahedral elements ,using a dedicated ANSYS Turbogrid software. The amount of layers and elements mesh data are shown in **table 2**.Sufficiently fine grid elements are created in the blade tip clearance region, around the blade, and at the hub and shroud walls. Sufficient mesh quality checks were performed by keeping the parameters like mesh angle and determinants within acceptable limits hence ,a grid independence study was carried out to ensure that the numerical solution are grid independent. Hence a fine grid size of elements was used for the CFD simulations reported in this paper ,the grid independence test as shown in **figure 3** .

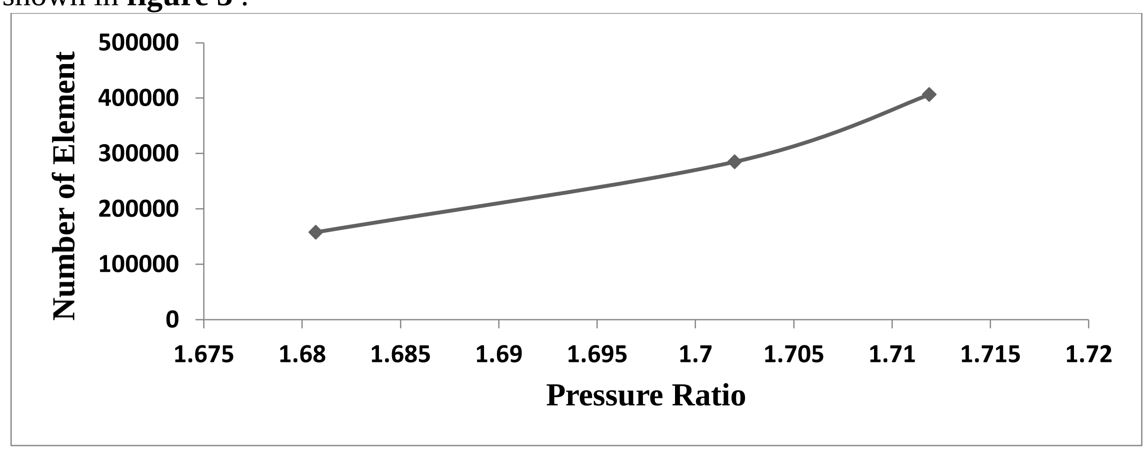


Figure 3: Grid Independence Test

Table 2: CFD Mesh Data of Air Ideal Gas

	No. of layers	No. of Nodes	No. of Elements
Stator	2	520175	488074
Rotor	3	434171	406179
Total	-	954346	894253

Grid size plays an important role in both convergence and accuracy of the solution. A coarse mesh is initially used to quickly examine the solver settings and boundary conditions. The grid generated should be appropriate to capture the complex flow phenomena like boundary layers, flow separation, leakage flows and secondary vortices in the blade passage. To capture all these flow characteristics, the number of cells should be large enough. But, as the number of cells increases, the computation time also increases rapidly. The grid size is, therefore, a compromise between computational time and accuracy of the results. Finer grids, in general, make the solution independent of the grid size and yield more accurate results but always require lager computational resources and time. Thus, a compromise between the grid size on one hand and convergence and accuracy on the other hand is required. Hence, a grid independence study was carried out to ensure that the numerical solutions are grid-independent. Hence, a fine grid size of elements was used for the CFD simulations. **Figure 4** shows the surface grid for the computational domain carried out by ANSYS CFX software.

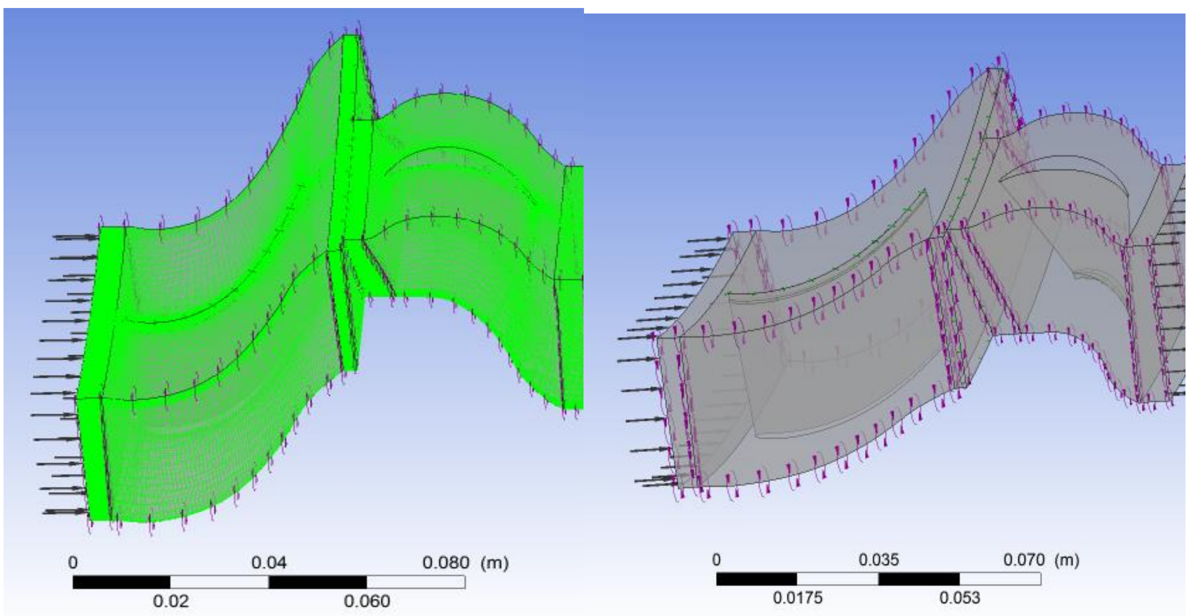


Figure 4: Surface grid for the computational domain

3.2 Fluid Flow Modelling

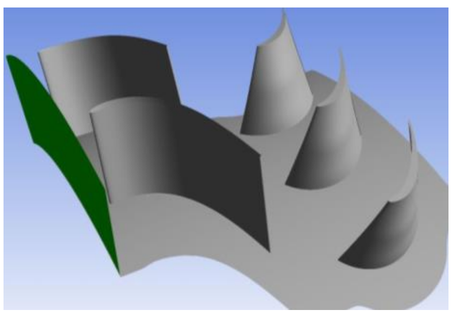
The Parametric computations are performed in a three dimensional-Turbulent CFD to obtain the performance of blade to blade configurations. The fluid zone comprises of one area enveloping all the rotating parts (blades and hub of rotor) and the other area of the stationary parts (shroud, inlet, outlet and stator nozzle). The left and right boundaries are defined as periodic . Turbulence is modelled using the k-ω-SST model. This model is a zonal combination of k-ω near the wall, nominally in the boundary layer, and k-ε away from the walls. When the near-wall mesh is compatible with the wall-function approach, this model behaves predominantly as a high-Reynolds number k-ε formulation. All surfaces are treated as adiabatic.

3.3 Boundary Conditions Used in CFD Simulation

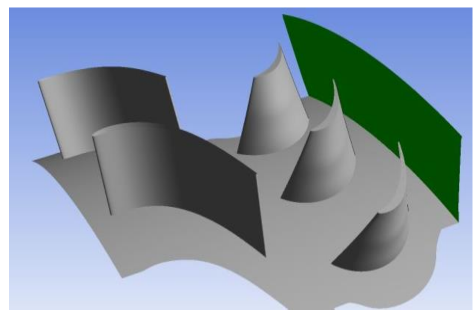
Boundary conditions specification is an essential part during CFD simulation .Calculation theory from symmetric to periodic, from wall to another, and from inlet to outlet . Total pressure and total temperature are applied to the inflow inlet boundary. The outflow outlet condition is set to static pressure. The boundary conditions used for an axial turbine design are shown in **figure 5** and listed in **table 3**.

Table 3: Boundary Conditions Used in CFD Calculations(Layth,2016).

Total temperature at domain inlet (Kelvin)	850
Total pressure at domain inlet (bar)	3.45
Flow angle at domain inlet (degree)	Axial Direction
Rotational speed (rpm)	30000
Exit static pressure (bar)	1.03
Physical Timescale	0.0002
Stator’s Outlet-Rotor’s Inlet	General Connection (Stage)
Flow Direction	Cylindrical (1,0,0)

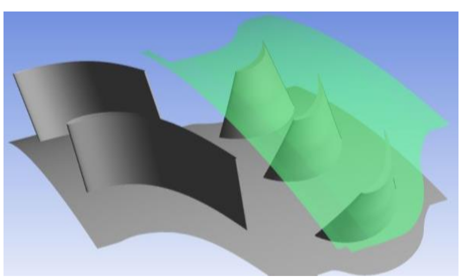


Inlet: The Pressure and Temperature are given

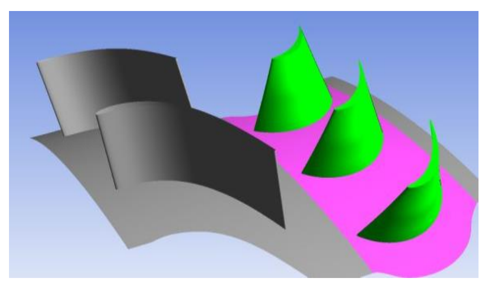


Outlet: Static pressure is given

Fig. 5 (a)



Shroud: which is a wall stationery boundary



Rotor(blade and hub):which is the wall moving boundary

Fig. 5(b)

Figure 5: Boundary conditions specification[Layth,2014].

4. Results and Discussion

The numerical method used by the solver part of the software requires an iterative process in order to obtain a solution. In general, the residual magnitude should decrease as the solution converges. When the magnitude of the residuals for all the quantities falls below the convergence level, the solver will stop iterating, and the results will be exported for post-processing. In **table 4** the overall performance prediction for the designed an axial turbine machine when running at the design point as indicated from the CFD results.The results obtained from calculations were compared with experimental work data which gave a good agreement for validation (Layth *et.al.*, 2016).

The blade loading is checked to make sure that the flow is not forced to reverse direction near the trailing edge. **Figure 6** shows the blade loading through the rotor passage at 50 % span. The red circle is the region where the flow is usually forced to reverse direction because the pressure at suction surface becomes larger than that on the pressure surface. This pressure reversal causes losses in the turbine, so in this design this reversal in the pressure can be avoided.

Figure 7 shows the velocity contours and stream lines at different span of the turbine rotor and stator. Separation is detected at the design point; the stator blade angle and number of rotor blades are iterated a number of times to avoid flow separation which is a major source of loss. The normal behavior of the fluid passing through an rotor blade can be seen, which is a low velocity flow in the pressure section of the blade and a high velocity flow on the suction section of the blade. The velocity vector magnitude at the mid Spanwise location for stator- rotor configurations from the stator inlet and rotor exit becomes uniform. Therefore, the flow losses decrease while the efficiency and pressure ratio becomes higher.

In addition, the contour of Mach number and pressure at different Spanwise location for an axial stator and rotor configurations from the stator inlet to rotor exit, as an example of CFD computations as shown in **figure 8**. The flow can be seen at the inlet of the designed type and its uniform distribution of the throat area between blade to blade passages. The flow can be seen in the space area between the trailing edge of the stator blade, and the leading edge of the rotor blade turbine which is close to Mach one meaning the space area ratio and the angle of the leading edge(55) of rotor blade which is a very important factor to modify in order to remove any choking of the flow for all configurations. It is clearly seen that the effect of flow stator angle to increase and decrease the backflow vortex. High-pressure can be noted on the stator leading edge due to the flow impact with the blade which is a normal behavior.

Figure 9 shows the Mach number difference across the streamwise location, from the inlet stator nozzle to the outlet rotor; it is visibly seen that Mach number is higher at the space between the stator exit and rotor inlet; therefore, it is very important to take into account the impact of a space ratio in the design. Computational fluid dynamic models give a much deeper understanding of the flow inside an axial turbine stator nozzle, enabling us to solve many problems easily and rapidly.The numerical analysis is carried out including the one stage stator –rotor flow passage only. The modification of a previous design of an axial turbine blades gives a better performance or a wide operating range.

Table 4: Overall Performance Results

Shaft Power (W)	610724
Inlet Flow Coefficient	0.4175
Total Pressure Ratio	1.7119
Total Temperature Ratio	1.0708
Nozzle Loss Coefficient	1.6383
Nozzle Efficiency %	77.6841
Total-to-Total isen. Efficiency%	80.1927
Total-to-Total poly. Efficiency%	79.3039

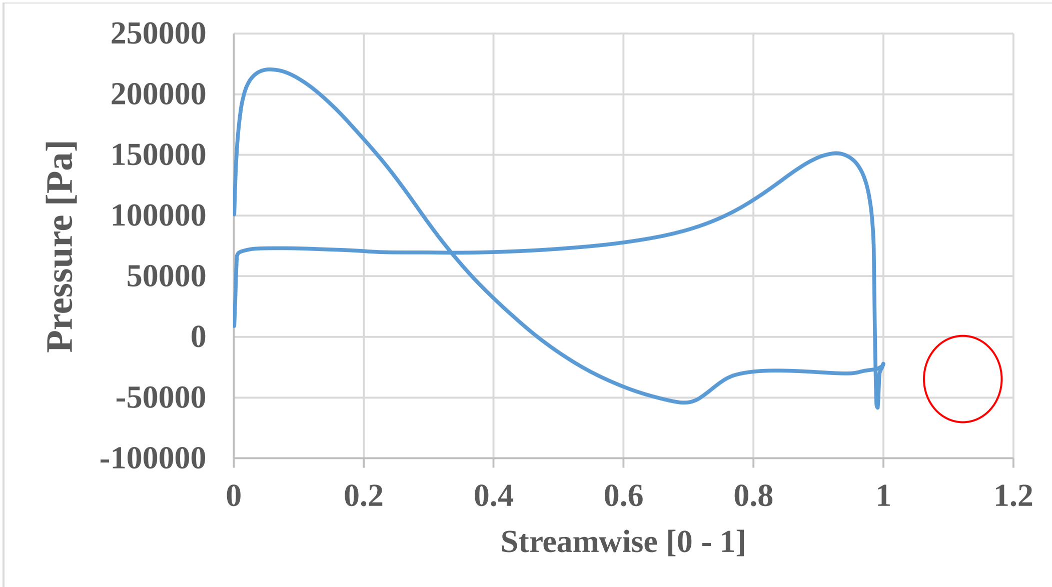
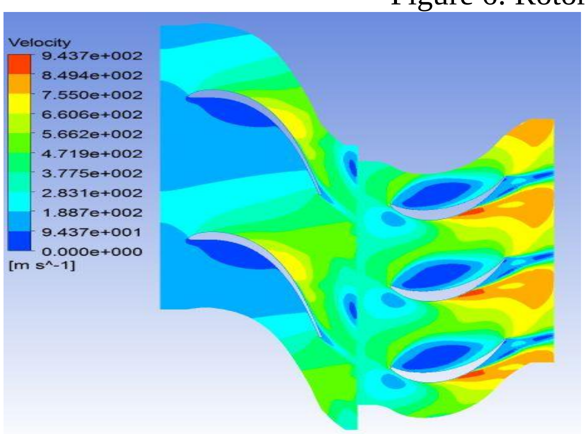
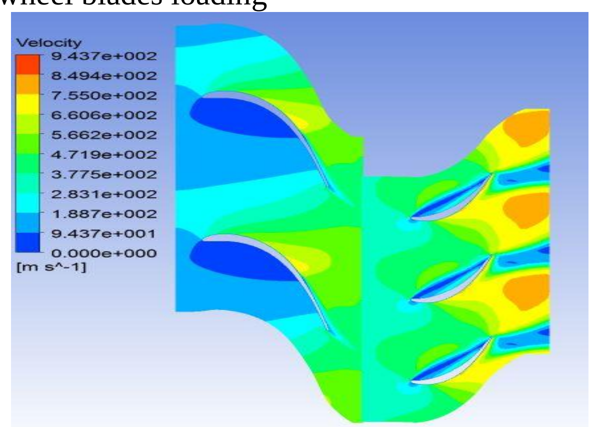


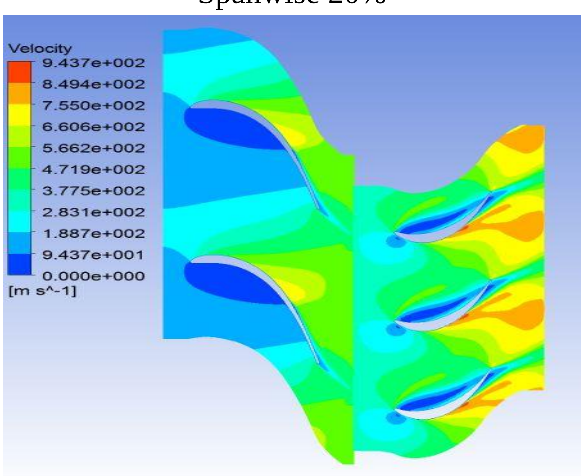
Figure 6: Rotor wheel blades loading

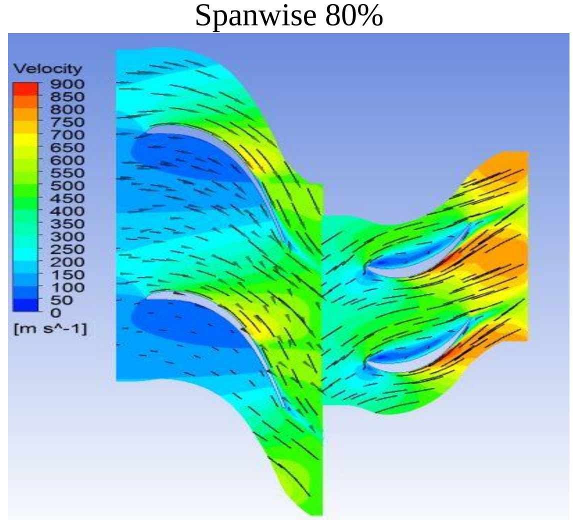


Spanwise 20%



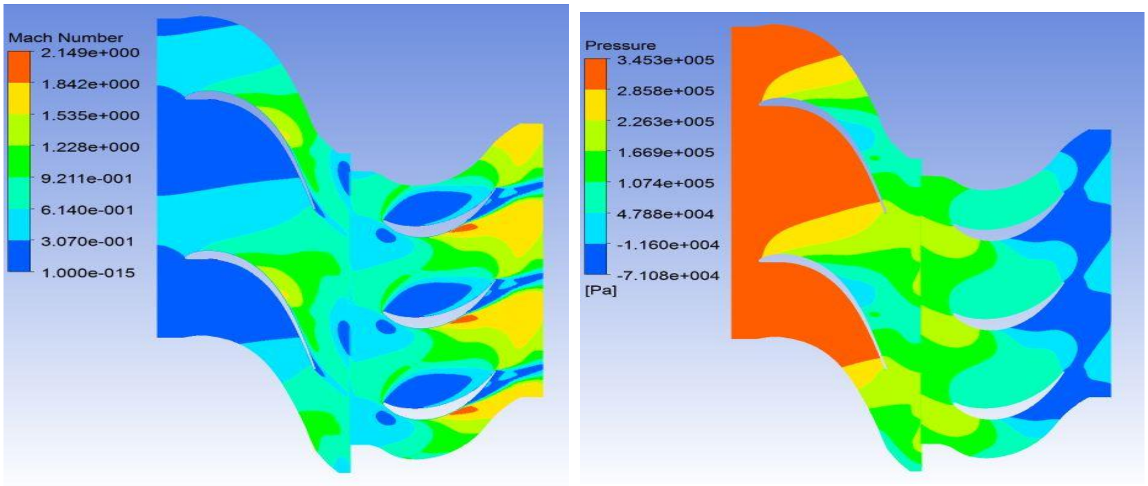
Spanwise 80%



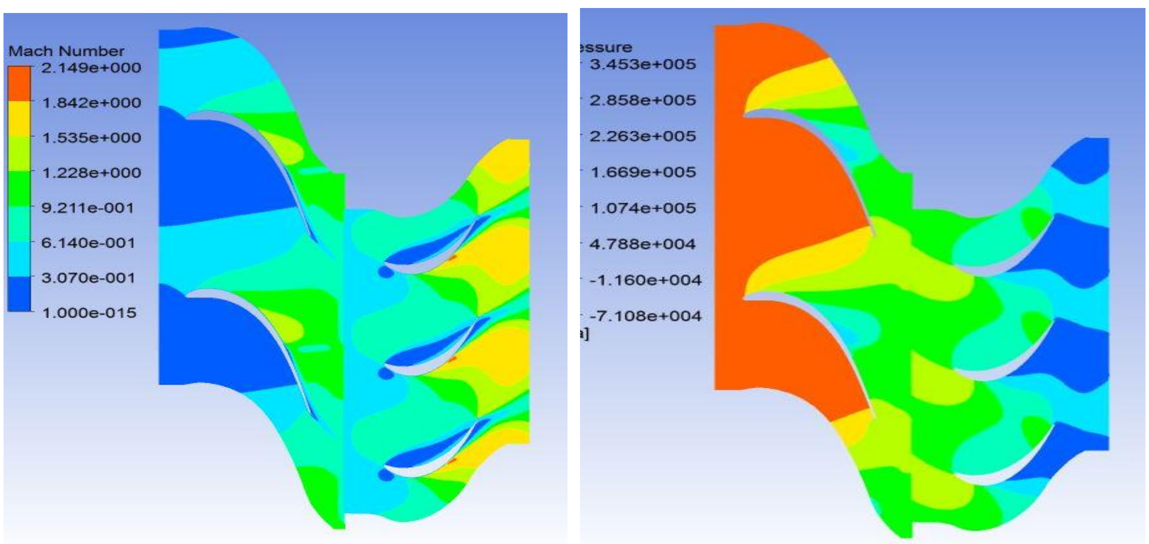


Spanwise 50%

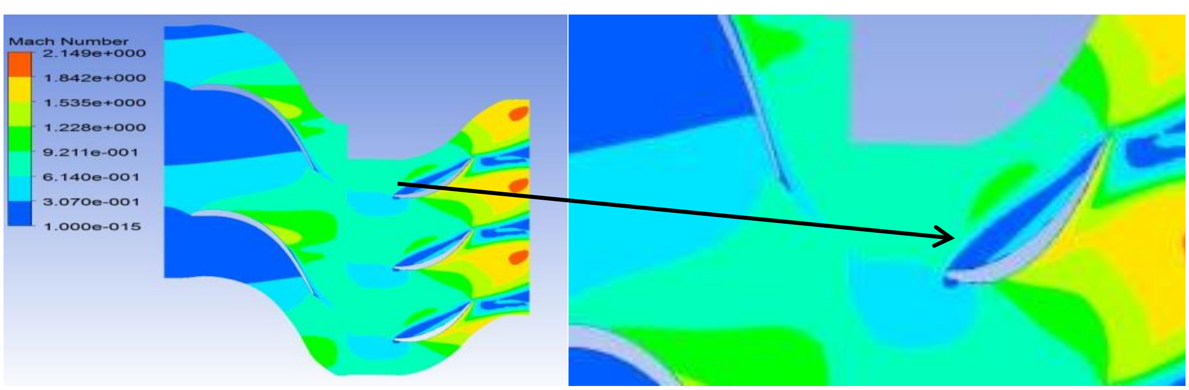
Figure 7: Velocity Contours& Stream Lines for Different Span

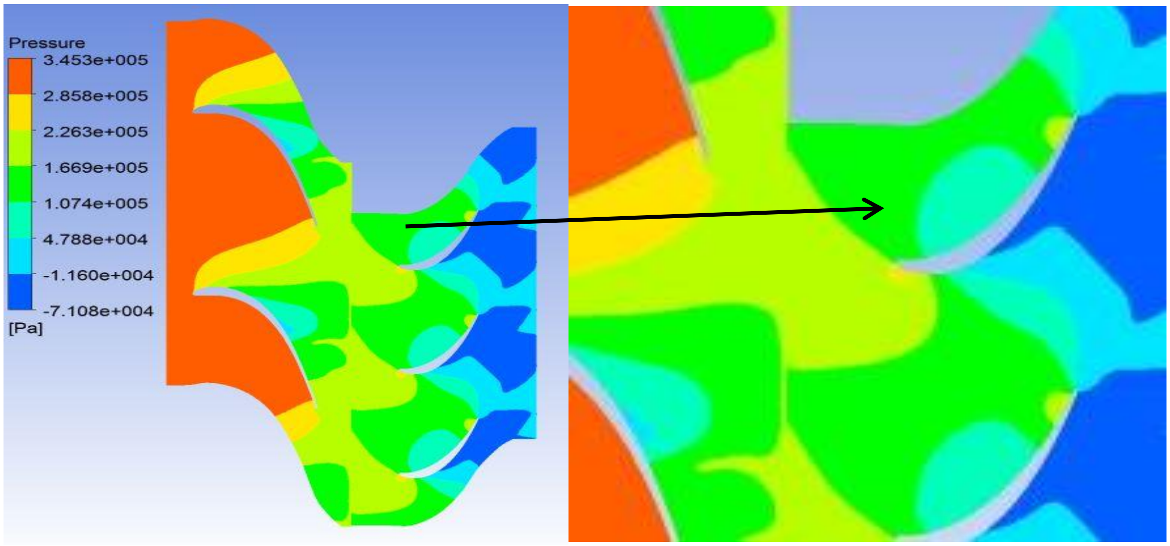


Spanwise 20%
Fig. 8(a)



Spanwise 50%
Fig. 8(b)





Spanwise 80%
Fig. 8(c)

Figure 8: Mach number and Pressure Contours for Different Span

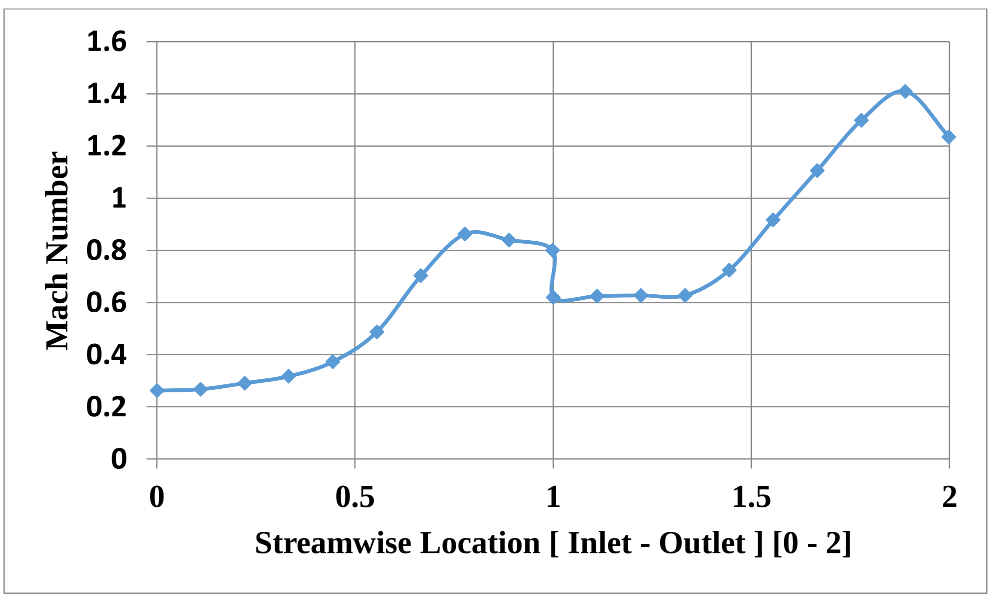


Figure 9: Mach number with Streamwise Location from Inlet to Outlet

5. Conclusions

The result is of relevance in internal combustion engine boosting applications. Turbocharger development is a major concern in current automotive engines because the downsizing trends. However, the matching between an axial rotor and stator blades is difficult, thus computational fluid dynamics (CFD) play a greater part in the aerodynamic design of turbomachinery than it does in any other engineering application. For many years in the design of a modern turbine or compressor has been unthinkable without the help of CFD and this dependence has increased as more of the flow becomes amenable to numerical prediction. The benefits of CFD range from shorter design cycles to better performance and reduced costs and weight. Steady state flow simulations have been conducted in order to analyse, the effects of the interaction stator –rotor on the axial turbine performance were systemically simulated and analysed. The analysis of the flow characteristics is also performed to obtain a better understanding of the blade to blade stator-rotor turbine flow behaviours, which at a certain speed is 30k RPM. The following conclusions are obtained; firstly, CFD simulation can predict and enhance the performance of an axial turbine. Secondly, the flow in the axial direction of the stator meridional passage is squeezed. The backflow and vortex near the pressure surface gradually slightly disappear. The flow tends to be uniform; hence the efficiency and total pressure ratio are both predicted. An effort is made to model the flow from the inlet to the exit of an axial turbine stage consisting of all the components using CFD tools. The vector plots and contour plots were generated for better understanding of fluid flow through the axial turbine stage. Conclusively, the aerodynamic results show that the performance is significantly affected by meridional passage of a rotor-stator turbine blades.

References

Al Jubori A., Al-Dadah R.K., Mahmoud S.,and Khalil K.M., 2016. Development of Efficient Small Scale Axial Turbine for Solar Driven Organic Rankine Cycle. ASME Turbo Expo 2016, pp. V003T25A011, doi:10.1115/GT2016-57845.

Bontempo R., Cardone M., Manna M., and Vorraro G. 2014. Highly Flexible Hot Gas Generation System for Turbocharger Testing. Energy Procedia (2014), Vol. 45, pp. 1116{1125. issn: 1876-6102. doi: 10.1016/.egypro.

Bontempo R., Cardone M., Manna M., and Vorraro G., 2015. Steady and unsteady experimental analysis of a turbocharger for automotive applications. Energy Conversion and Management, Vol. 99, pp 72-80, 2015 doi:10.1016/j.

Filsinger D., Frank C., and Schafer O., 2005. Practical Use of Unsteady CFD and FEM Forced Response Calculation in the Design of Axial Turbocharger Turbines. ASME Turbo Expo 2005, pp. 601-612, doi:10.1115/GT2005-68439.

Filsinger D., Szwedowicz J., and Schäfer O., 2001. Approach to Unidirectional Coupled CFD – FEM Analysis of Axial Turbocharger Turbine Blades. ASME Turbo Expo 2001: Power for Land, Sea, and Air, pp. V004T03A047, doi:10.1115/2001-GT-0288.

Kibsey M.D., and Sjolander S.A., 2016. Influence of Mach Number on Profile Loss of Axial-Flow Gas Turbine Blades. ASME Turbo Expo 2016, pp.V02BT38A015.

Layth H. Jawad, Abdullah S., Zulkiflil R. and Mahmood W.M.F.W., 2014.Numerical Study on the Effect of Interaction Vaned Diffuser with Impeller on the Performance of Modified Centrifugal Compressor. Journal of Mechanics, Cambridge University Press, the Society of theoretical and Applied Mechanics.Vol.30,No.2,pp.113-121.

Layth H. Jawad, Mahir H. Majeed and Boriere Ahmed, 2016.Numerical Investigation of Performance Improvement of a Modified Turbocharger Centrifugal Compressor with second splitter and vaned diffuser. Journal of Karbala University,No.228

Leonard T., Spence S., Early J. and Filsinger D., 2013. A numerical study of automotive turbocharger mixed flow turbine inlet geometry for off design performance. Materials Science and Engineering ,volume 52 , Issue 4, pp 042012 , doi:10.1088/1757-899X/52/4/042012 .

Liu Y. B., Zhuge W. L., Zhang Y. J. and Zhang S. Y., 2016. Numerical analysis of flow interaction of turbine system in two-stage turbocharger of internal combustion engine. Materials Science and Engineering (2016), Volume 129, Issue 1, pp 012004.

Marelli S., Marmorato G., Capobianco M., and Boulanger J., 2016. Towards the Direct Evaluation of Turbine Isentropic Efficiency in Turbocharger Testing. SAE Technical Paper 2016-01-1033, 2016, doi:10.4271/2016-01-1033.

Morrison R., Spence S., Kim S., Filsinger D., & Leonard T., 2016. Investigation of the Effects of Flow Conditions at Rotor Inlet on Mixed Flow Turbine Performance for Automotive Applications. International Turbocharging Seminar 2016, Tianjin, China.

Padzillah M.H., Yang M., Zhuge Z. and Martinez- Botas R.F., 2014. Numerical and experimental investigation of pulsating flow effect on a nozzeld and nozzleless mixed flow turbine for an automotive turbocharger. Proceedings of the ASME Turbo Expo 2014, Germany , pp V02DT42A027.

Padzillaha M.H., Rajoo S., Yang M. and Martinez-Botas R.F., 2015 .Influence of pulsating flow frequencies towards the flow angle distributions of an automotive turbocharger mixed-flow turbine. Energy Conversion and Management. Volume 98, 1 July 2015, Pages 449–462 .

Pan Y., Yuan Q., Chen Q., Ge Q., and Ji D., 2016 . CFD Analysis of the Unsteady Flow in a Two-Stage Axial Turbine. ASME Turbo Expo 2016, pp. V02BT38A021.

Patel B. N. and SUBHEDAR D.,2016.Experimental and Numerical Investigation of Diesel Engine Turbocharger. International Journal of Automobile Engineering Research and Development ISSN(P):2277-4785;ISSN(E): 2278-9413 Vol. 6,Issue 3.

Rajeevalochanam P., Sunkara S., and Mayandi B., 2016. Design of Highly Loaded Turbine Stage for Small Gas Turbine Engine. ASME Turbo Expo 2016, pp. V02CT39A005, doi:10.1115/GT2016-56178.

Roclawski1 H., Gugau M. and Böhle M., 2016. CFD Analysis of a Radial Turbine during Load Step Operation of an Automotive Turbochargers. International Symposium on Transport Phenomena and Dynamics of Rotating Machinery. Hawaii, Honolulu.

Salameh G.,Chesse P.,Chalet D.,and Talon V., 2016. Experimental Study of Automotive Turbocharger Turbine Performance Maps Extrapolation.SAE Technical Paper 2016-01-1034, 2016, doi:10.4271/2016-01-1034.

Straka P., 2016. Numerical simulation of high-swirl flow in axial turbine stage**.** EPJ Web of Conferences, 2016**,** Volume 114, pp 02115.

Zhao R., Zhuge W. and Zhang Y. 2016. Numerical study of a two-stage turbine characteristic under pulsating flow conditions. Journal of Mechanical Science and Technology, Volume 30, Issue 2, pp 557–565.